# HEAT TRANSFER PROBLEMS AND THERMAL STRESSES

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x} \left( \lambda_x \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \lambda_y \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( \lambda_z \frac{\partial T}{\partial z} \right) + q_v(x, y, z, t) ,$$

where: T(x, y, z, t) - temperature,  $q_v$  - heat generation (W/m<sup>3</sup>),  $\rho$  - density (kg/m<sup>3</sup>),  $\lambda_x, \lambda_y, \lambda_z$  - thermal conductivity (W/mK), c - specific heat (J/kg).

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most materials do vary with temperature, so the analysis usually is nonlinear.

**Transient thermal analysis determines** temperatures and other thermal quantities that vary over time. Many heat transfer applications--heat treatment problems, nozzles, engine blocks, piping systems, pressure vessels, etc.--involve transient thermal analyses.

Temperature distribution that a transient thermal analysis calculates is often used as input to structural analyses for thermal stress evaluations.

A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time. To specify time-dependent loads the load-versus-time curve should be divided into load steps.

If you use individual load steps, each "corner" on the load-time curve can be one load step, as shown in the following sketches.



Examples of load-versus-time curves - stepped and ramped loads

For each load step, you need to specify both load values and time values, along with other load step options such as stepped or ramped loads, automatic time stepping, etc. You then write each load step to a file and solve all load steps together.

### Temperature-dependent coefficient of thermal expansion $\alpha_t(T)$

If  $\alpha_t$  is the thermal expansion coefficient then the typical component of thermal strain is  $\epsilon_{th} = \alpha_t (T) (T-T_0)$ .

If  $T_0 = T_{ref}$ , where  $T_{ref}$  is the reference temperature at which zero strains exist such a coefficient is correctly used. If this condition is not true an adjustment must be made (MPAMOD command in the Preprocessor).

#### **Thermal stresses**

Thermal stresses usually are analysed using sequential method.

The *sequential method* involves two or more sequential analyses, each belonging to a different field. You couple the two fields by applying *results* from the first analysis as *loads* for the second analysis. In the case of thermal-stress analysis the nodal temperatures from the thermal analysis are applied as "body " loads in the subsequent stress analysis

Ansys enables also using *direct method* which involves just one analysis that uses a *coupled-field element type* containing all necessary degrees of freedom.



Data flow for a sequential coupled field analysis.

### Example 1 - thermal stresses in steady state heat flow

In the steel thick pipe we have the internal temperature  $T_w=100^{\circ}C$  end external temperature  $T_z=20^{\circ}C$ . The inner radius is a=30mm, and outer b=40mm. Show the temperature distribution, von Mises stress and stress components in the cylidrical coordinate system. E=2e11Pa, v=0.3,  $\alpha_t=1.2e-5$  1/K, k=50 W/mK. Consider the pipe constrained in the axial direction at both ends.

Analytical solution (plane strain problem,  $T_{ref}=0^{\circ}C$ ):



$$T(r) = T_w + \frac{T_z - T_w}{\ln\left(\frac{b}{a}\right)} \ln\left(\frac{r}{a}\right)$$

$$\sigma_r(r) = C \left[ \ln\left(\frac{b}{r}\right) / \ln\left(\frac{b}{a}\right) - \left(\frac{b^2}{r^2} - 1\right) / \left(\frac{b^2}{a^2} - 1\right) \right]$$

$$\sigma_t(r) = C \left[ \left( \ln\left(\frac{b}{r}\right) - 1\right) / \ln\left(\frac{b}{a}\right) + \left(\frac{b^2}{r^2} + 1\right) / \left(\frac{b^2}{a^2} - 1\right) \right]$$

$$\sigma_z(r) = v \left(\sigma_r + \sigma_t\right) - \lambda ET$$

$$C = \frac{-E\alpha_t \left(T_w - T_z\right)}{2(1 - v)}$$

T<sub>z</sub>

From the above formulas we have e.g.  $\sigma_t$  (a)=-150.2 MPa,  $\sigma_t$  (b)=124.1 MPa.





TEMP

Temperure along the thickness of the pipe



Radial stress (Sx) ,hoop stress (Sy) , axial stress (Sz) and von Mises stress (Sred) along the thickness of the pipe (MPa)



Distribution of radial stress (Sx) ,hoop stress (Sy) , axial stress (Sz) and von Mises stress (Sred) Cylindrical coordinate system

### Approach

The analysis may be performed using plane strain model (cross-section of the pipe), axisymmetric model, or 3D model. In each case only a segment of the pipe may be analysed with the adequate symmetry conditions.

### Summary of steps in numerical analysis (3D):

#### Preprocessor

-define geometry of the analysed region (part of the cylinder),

-define the material properties  $(E,v,\lambda,k)$ 

-define the element type (thermal solid)

-specify the boundary conditions (temperatures  $T_w$  and  $T_z$  applied on the adequate areas)



-mesh the model

#### Solution

-solve the current load step (Solve>Current Load Step)

### General Postprocessor

-review the results- plot the maps of interest and the temperature as the graph along the path (thickness).

### Preprocessor

- change the element type from *thermal solid* to *structural solid* (*Element Type>Switch* Element Type)

- switch element technology to Enhanced Strain Preprocessor

	ent Type		
🖬 🗛	dd/Edit/De	elete	
🖬 S	T Element	Types	X
🖬 🗛			
🖬 R			
📼 E	Define	d Element Types:	
⊞ Rea	Туре	1 SOLID185	
⊞ Mat			
Sec		A SOLID185 element type options	
⊞ Moc		Options for SOLID185, Element Type Ref. No. 1	
. Mes			
⊞ Che		Element technology K2	Full Integration
. Nun ₪		Layer construction K3	Full Integration
Arcl			Reduced integration Enhanced Strain
Cou		Element formulation K6	Simple Enhanced Strn
FLO			
Mult		User defined initial stress K10	No USTRES routn 💌
Loa			
Phy		OK Cancel	Help
Patł			

#### Solution

- apply the boundary conditions for stress analysis (*Define Loads>Displacements> Symmetry B.C.>On Areas*)

- apply the nodal tempertatures as a load in stress analysis (file *jobname.rth* from thermal analysis)



- solve the current load step

#### General Postprocessor

- review the results - plot the stresses as graphs along the path (thickness). Use cylindrical coordinate system: (*Options For Output>Results Coordinate System*)

### **Tasks and questions:**

1. Repeat the analysis using 2D plane strain or axisymmetric model. Compare the obtained results with the results corresponding to the 3D model.

2. Perform the adequate analysis for the pipe with unconstrained ends (without the axial compression). Explain the differences.

3. Find the results corresponding to point 2, but with the 10mm insulation (E=1\*10<sup>3</sup> MPa, v=0.35, k= 0.1 W/m<sup>2</sup>K)

3. Repeat the calculations using the model of convection b.c. : At the int. surface : bulk temp. 100C, film coefficient  $h=500W/(m^2K)$ At the ext. surface: bulk temperature 20C, film coefficient is  $h=10W/(m^2K)$ . Explain the results.

In each case save the results: FE mesh, temperature distribution and stress components distributions in the form of contour plots and graphs along the path (thickness).

### Example 2 - thermal stresses in transient heat flow

Steel balls of diameter d=12mm are heated to  $T_1$ =850°C and then quenched in oil. The temperature of the oil is assumed as constant  $T_0$ =40°C. Heat exchange coefficient at the surface oil – the ball is h=400W/(m<sup>2</sup>K).

How long should the balls stay within the oil bath to get the temperature  $T_2=100^{\circ}C$  in the centre?

What is the maximum von Mises stress during the process?

Material properties of the steel : density  $\rho$ =7800kg/m<sup>3</sup>, specific heat c=444J/(kgK), thermal conductivity k=40W/mK, coefficient of thermal expansion  $\alpha_t$ =1.2e-5 1/K, Young modulus E=2e11Pa, Poisson's ratio v=0.3.



b) the difference DT=Tw-Tz



Temperature distribution (°C) and von Mises stress (Pa) after 1 s of cooling



Temperature distribution (°C) and von Mises stress (Pa) after 48 seconds of cooling.

# Approach

The analysis may be performed using axisymmetric or 3D model. In each case only a segment of the ball may be analysed with the adequate symmetry conditions.

### Summary of steps in numerical analysis (3D):

Preprocessor

-define geometry of analysed region (part of the sphere by intersection of sphere and block),

-define material properties (E,v, $\lambda$ ,k,  $\rho$ ,c)

-define element type (thermal solid)

-specify the boundary conditions (convection on external surface with *bulk temperature*  $40^{\circ}$ C, *film coefficient* 400W/m<sup>2</sup>K)



-mesh the model

#### Solution

- set type of analysis - ANALYSIS TYPE - transient (solution)-



 set initial condition TREF= 850°C (being the initial temperature)
 set time, time step and related parametrs (Load step options), e.g.: time=50sec, time step 0.2 sec., stepped, auto time step OFF

ANSYS Main Menu	۱		
Preferences			
Preprocessor     Solution			
<ul> <li>Solution</li> <li>Analysis Type</li> </ul>			
🖬 Fast Sol'n Optn		Time and Time Step Options	
Define Loads		Time and Time Step Options	
Load Step Opts			
🗉 Output Ctrls		[TIME] Time at end of load step	50
Solution Ctrl			
Time/Frequenc		[DELTIM] Time step size	250
🧰 Time - Time Step		[VD 0] Observed an environment la se	1
🔲 Time and Substps		[KBC] Stepped or ramped b.c.	
🖬 Damping			Ramped
<ul> <li>Time Integration</li> <li>Nonlinear</li> </ul>			e rampou
			Stepped
⊞ Other			
Reset Options		[AUTOTS] Automatic time stepping	
🔜 Read LS File			O ON
🔜 Write LS File			0 ON
🗉 Initial Stress			OFF
Physics			
🗉 SE Management			Prog Chosen
Results Tracking Solve		[DELTIM] Minimum time step size	
⊞ SOIVE ⊡ FLOTRAN Set Up		[Deering minimum diffe step size	
Run FLOTRAN		Maximum time step size	
⊞ FSI Set Up		Maximum and step size	
MultiField Set Up		Use previous step size?	Yes
ADAMS Connection		Ose previous step size:	I∙ res
Diagnostics		[TCREC]. Time step yeast based an enseife time points.	
🖬 Abridged Menu		[TSRES] Time step reset based on specific time points	

output controls (file write sequency to every substep or every Nth step)

ANSYS Main Menu 🛞		
🗟 Preferences		
Preprocessor		
Solution		
Analysis Type	A Calution Drinter & Cantuals	
Define Loads	Solution Printout Controls	
🗆 Load Step Opts	[OUTPR] Solution Printout Controls	
<ul> <li>Output Ctrls</li> <li>Solu Printout</li> </ul>	These These for a single sector 1	
Grph Solu Track	Item Item for printout control	Basic quantities 📃 💌
Show Status	FREQ Print frequency	
B PGR File	The quality	
Incl Mass Summry		🔘 None
Integration Pt		
🗄 Other		🔿 Last substep
Stop Solution		Every substep
Reset Options		C Livery substep
🔜 Read LS File		Every Nth substp
🔜 Write LS File		+
Initial Stress	Value of N	2
E Management		
Results Tracking E Solve	(Use negative N for equally spaced data)	
⊞ FSI Set Up		
MultiField Set Up	Cname Component name -	All entities 🔹 👻
ADAMS Connection	for a substate when a substate is the last successive of	,
Diagnostics	- for which above setting is to be applied	
Unabridged Menu		
∃ General Postproc		
⊞ TimeHist Postpro	OK Apply	Cancel Help
🗄 Topological Opt		

-solve Current Load Step

# General Postprocessor

Read the results corresponding to the chosen time point

ANSYS Main Menu 🤇 🤇	8				
🗟 Preferences					
Preprocessor	SET,LIST (	Command			
🗄 Solution		Johnnand			
🗆 General Postproc	\Lambda Results Fil	e: kulka.rth			
🖬 Data & File Opts					
Results Summary	Available Da	ata Sets:			
Read Results	Set	Time	Load Step	Substep	Cumulative
🔤 First Set					
🖩 Next Set 🗟 Previous Set	1	0.20000	1	1	1
🔤 Previous Set	2	0.40000	1	2	2
By Pick	3	0.60000	1	3	3
By Load Step	4	0.80000	1	4	4
By Time/Freq	5	1.0000	1	5	5
By Set Number	6	1.2000	1	6	6
ELOTRAN 2.1A	7	1.4000	1	7	7
Plot Results	8	1.6000	1	8	8
🗉 List Results	9	1.8000	1	9	9
Query Results	10	2.0000	1	10	10
Options for Outp			-	20	
Results Viewer	11	2.2000	1	11	11
Write PGR File	12	2.4000	1	12	12
⊞ Nodal Calcs ⊞ Element Table	13	2.6000	1	13	13
	14	2.8000	1	14	14
Surface Operations	15	3.0000	1	15	15
E Load Case	16	3,2000	1	16	16
Check Elem Shape		Read			Next
Write Results					
ROM Operations		$\searrow$			
Submodeling					
🗄 Fatigue		C	ose		
Define/Modify					
Nonlinear Diagnostics	19 3	. 8000	1	19	19
E Darah			-		

-review the results corresponding to the arbitrary point of time -animate the results (General Postprocessor – PlotCTrl)

# TimeHist Postprocessor

-review the results e.g. variation of temperature with respect to time at chosen points (commands *Define Variables i Graph Variables*):

SAVE_DB RESUM_DB	QUIT POWRGRPH ANSYSWEB TU	UIT POWRGRPH ANSYSWEB TUTORIAL WHATSNEW				
	\Lambda Time History Variables - aug	j28a.rth				
ANSYS Main Menu	File Help					
<ul> <li>Preferences</li> <li></li></ul>	Yariable List	None 💌 💊 🕸				
General Postpro	Name Element Node	Result Item				
TimeHist Postpro     Zariable View	TIME	ime-History Yariable				
		t Item				
📰 Store Data	Resul	r rrem				
📰 Define Variabl	F 🔂	Favorites				
Read LSDYNA	n 🖉 1	Nodal Solution				
List Variables		💋 DOF Solution				
Graph Variable		🚽 🎓 Temperature				
🕀 Math Operatic	<b>T</b>	🛜 Thermal Gradient				
🕀 Table Operati	Calculator	🦳 Thermal Flux				
Smooth Data		Element Solution				
Generate Spe						
Topological Opt						
ROM Tool	() Resul	t Item Properties				

-find the difference between the temperature in the centre of the ball and the surface – delta T:

ile Hel						
±×	i 🗖 🗖 🖻	1 🖻 📙	None 🗾 💊 🕸			
Variable	List					
Name	Element	Node	Result Item	Minimum	Maximum	X-Axis
IME			Time	0.05	50	$\odot$
zew		22	Temperature	87.6881	842.994	0
ſwew		2	Temperature	89.1317	849.959	0
deltaT			Calculated	1.44368	22.8599	0
4	or					
∢   Calculat	or deltaT	= {T	vew}-{Tzew}			

-show the defined functions in the form of graphs

#### Preprocessor

-change the element type from *thermal solid* to *structural solid* 



-apply the boundary conditions for stress analysis (symmetry)

-apply the nodal tempertatures as a load in stress analysis at the arbitrary point of time  $t_{arb}$  (file *jobname.rth*)



#### Solution

- set analysis type to static

-set time=1, analysis using 1 substep (Time/Frequency) -solve Current Load Step

#### General Postprocessor

-Review the results for the time t<sub>arb</sub>

In the presented way of analysis the thermal stresses corresponding to the chosen time  $t_{arb}$  were presented. If we want to find the history of stresses in a period of time the corresponding history of loads for structural analysis should be built in the form of Load Step files (Files *Jobname.s i*, where i is the number of the load step).

It may be also performed automatically using for example the presented below macro, which is set of commands written using Ansys Parametric Design Lenguage (APDL). The file may be read by the program in Solution phase using the INPUT command.

```
/COM, Creation of files jobname.si for history of linear thermal stresses
/COM, The commands may be input after changing elements to structural and
/COM, after applying the adequate structural boundary conditions
     *ask,case,name of the file rth (jobname) =,'file'
     *ask,time_in,initial time =,1
     *ask,time e,final time =,1
     *ask,liczba_p,number of time segments = ,1
     przyr=(time e-time in)/liczba p
     chwila=time in
     ANTYPE,0
     *DO,k,1,liczba p
     LDREAD, TEMP, , , chwila, , case, 'rth',
     TIME, chwila
     AUTOTS,0
     NSUBST,1, , ,1
     KBC,1
     LSWRITE, k,
     chwila=chwila+przyr
     *ENDDO
```

# Tasks and questions:

-Repeat the analysis using axisymmetric model. Compare the results.

-Find the influence of heat exchange coefficient and thermal conductivity on the results (maximum stresses and time of cooling)

-Perform the analysis using the elasto-plastic model of the material. Why in this case residual stresses may be expected?